A hybrid slurry CFD model: Euler-Euler to Euler-Lagrangian (in development)

Alasdair Mackenzie

Weir Advanced Research Centre, University of Strathclyde, Glasgow, Scotland
Outline

- Background, context and motivation to the problem
- Development of hybrid model will be explained
- Test case will be shown

- Tutorial can be found on Chalmers website (end of January): http://www.tfd.chalmers.se/~hani/kurser/OS_CFD_2016/
Background

- Weir group produce equipment for the mining and oil and gas industries

- Erosion is a large problem

- CFD modelling is used to predict erosion = better designs

- Longer pump life = happy customer :}

5th United Kingdom & Éire OpenFOAM® User Meeting
Impellers

Before

After

Could be as little as 2 weeks of continuous running for this to happen
Problem/Motivation

- Need particle impact data at the wall for erosion modelling
- Fluid/particulate flow simulation is computationally expensive: especially for dense slurries
- Solution to make faster: Combine with two-fluid model

Velocity contours of submerged jet impingement test

Dotted region where particles are necessary for impact data
A simple geometry was chosen for solver development

- reactingTwoPhaseEulerFoam for Euler-Euler
- DPMFoam for Euler-Lagrange
- OpenFOAM 3.0.x was used

Geometry shown with sizes in metres
Description of Solvers

reactingTwoPhaseEulerFoam

Euler-Euler

Two fluid model

Both phases treated as continuum

Incompressible model: setting in dictionary

Fast to solve

DPMFoam

Euler-Lagrange

Fluid/particle model

Transient solver for coupled transport of kinematic particle clouds

Includes the effect of volume fraction of the particles on the continuous phase
Combining the solvers

- A new solver was made based on the EE model
- To have 2 solvers running, 2 regions were created
- To go from fluid to particles, we need a transition
- An outlet/inlet is needed for particle phase, but shouldn't affect the rest of the flow
- Solution…
Baffles + Regions

- createBaffles: makes internal surface into boundary face
- master and slave patch created
- splitMeshRegions: Splits mesh into 2 separate regions
- BC’s can now be applied to baffle patches
- chtMultiRegionFoam: Inspiration for solving regions sequentially
Interpolation

- **patchToPatchInterpolation**: transfers data between two patches
- All variables are interpolated: U1, U2, p, p_rgh, alpha1, alpha2, k, epsilon, nut, and theta
- After this is implemented, the domain runs as if it was one region, not two: the surface doesn’t affect the flow
- ‘back pressures’ are taken into account by interpolating upstream

**Iterative loop**

1. Solve in Region0
2. Interpolate from master patch to slave patch
3. Solve in Region1
4. Interpolate from slave patch to master patch

---

5th United Kingdom & Éire OpenFOAM® User Meeting
DPMFoam added

- Code from DPMFoam was added to new solver
- Particles injected from slave patch after back interpolation (slave to master)
- Particles are only in region1 (near wall)
- Injection values based on phase 2 from region0 by using a lookup table: kinematicLookupTableInjection
DPMFoam injection

- Modified kinematicLookupTableInjection used to inject particles

- Lookup table is updated every time step (but not read every time step: advice welcome!)

- 1 line = 1 cell (100 cells in this case)

- Values for particle injection are based on new updated values so solver can deal with geometry changes etc. See Lopez’ presentation for more details:


5th United Kingdom & Éire OpenFOAM® User Meeting
DPMFoam injection

- Number of parcels to be injected is calculated from volume flow rate of 2nd phase of fluid.

- Number of parcels/cell = (alpha particles * area of cell * normal velocity component to cell boundary face) / (volume of one particle * number of particles/parcel * number of time-steps/second)
Velocity contours

- 2D slice through Z normal. Particles injected from slave patch
Velocity contours
Comparison

- New solver was compared against standard EL and EE solvers
- Hybrid model is almost double the speed of the EL

Execution time from 0-0.39s: % mass concentration (MC)

<table>
<thead>
<tr>
<th>Model</th>
<th>Execution Time (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1% MC</td>
</tr>
<tr>
<td>Hybrid model</td>
<td>225</td>
</tr>
<tr>
<td>Euler-Lagrange</td>
<td>420</td>
</tr>
<tr>
<td>Euler-Euler</td>
<td>102</td>
</tr>
</tbody>
</table>
Comparison

Hybrid Model particle impacts

Data taken from bottom wall on pipe bend

DPMFoam particle impacts

5th United Kingdom & Éire OpenFOAM® User Meeting
Future work

- Validation of hybrid model: CFD and experimental (PIV)
- Particles to fluid, for after region of interest…
- Move lookupTable to memory?
- Make solver re-read the lookupTable (suggestions welcome)
Conclusion

- Solver should dramatically reduce computational time
- Particle data should still be present near walls, where required
- Enable better design of mining equipment

Worn impeller of slurry pump
Thank you. Questions?

alasdair.mackenzie.100@strath.ac.uk